RELEASE NOTES — AUTOTRAX Version 1.10a

There are many changes and enhancements included in Version 1.10a of AUTOTRAX. All users are urged to carefully read these release notes in conjunction with the Reference Manual.

TRAXEDIT Program

1. **Disk Organization** The AUTOTRAX program files should NOT be kept in the same directory as any other PROTEL files. Each PROTEL program uses a graphics driver called GRAPH.DRV. More than one program resident in a directory can cause the wrong driver to be selected.

2. **Graphics Drivers** supported in V 1.10:

   PARA800.DRV: Replaces VGA800 (a non-standard mode). This driver will work with Paradise VGA and other cards using the same chipset. The Genoa Enhanced EGA drivers also support cards using the same chipset (such as the TOP SEGA 800 card).

HERC	Hercules monochrome	348 x 720
CGA	Color Graphics Adapter	320 x 200 4 color
EGA	Enhanced Graphics Adapter	640 x 350 16 color
VEGA	Vega Deluxe Expanded EGA	640 x 480 16 color
GENOA640	Genoa Expanded EGA	640 x 480 16 color
GENOA800	Genoa Expanded EGA	800 x 600 16 color
GENOA912	Genoa Expanded EGA	912 x 350 16 color
VGA640	Video Graphics Array	640 x 480 16 color
PARA800	Paradise Expanded VGA	800 x 600 16 color

3. **SETUP/OPTIONS** Menu additions.

   a) Flash Cursor On/Off (default is Off).
   b) Question Delete On/Off (default is On).

4. **TRAXEDIT.DFT** file changes:

   a) The graphics driver name is now stored in this file. This allows changing drivers without losing all settings. Only color settings will return to the default values.

   b) The version number is in the file. If the version number changes the .DFT file will be ignored. There is no need to delete the .DFT file unless you wish to restore all settings to default.

5. **Status Line** Now includes current Track Size, current Pad Type and current Snap Grid size.

6. While using **HIGHLIGHT**, **ROUTE** or **MAKE NETLIST** you can now use the following commands: PgUp, PgDn, Home, End, Arrow keys, Zoom menu and Grid menu. The **ZOOM** and **GRID** menus may now be used during any command involving a cursor move eg: *Place/Track*, *Place/Component*, etc. The mouse is only supported in *Zoom/Window*.

7. **HIGHLIGHT** Menu addition: *Reset Highlight* turns off highlight.

8. **Screen Redraw** Hitting the space bar will now terminate screen redraws.

9. **EMS** The command *Make Netlist* and **Via Minimisation** were not working correctly under EMS in v 1.07. These have been corrected.

   If you have EMS installed then the 32K of DOS memory used to store the screen image under the menu is now saved and 32K of EMS is used instead.

10. **Power and Ground Planes**

    Care and forethought is required when planning the use of Power and Ground Plane inner layers. Adequate clearance must be maintained around all non-connected pin holes to ensure that shorts do not result from slight mis-alignment of layers. The recommended minimum for this clearance is 0.4mm (.016") for boards less than 305mm (12") in

a side. Larger boards require a clearance of at least 0.5mm (.028").

AUTOTRAX plots a copper-free area around non-connected pins **equal to the pad outside diameter**. Typical pin or lead diameters require a 0.7mm (.028) hole. Thus, a standard .062" pad will meet this requirement. These considerations must be applied to all pad/hole combinations used in your design.

You can change these clearances in a completed design by re-editing all pads prior to plotting the GND and PWR plane layers using the Global Edit facility.

11. Keyboard Macros

When editing an existing macro or defining a new one, the **Scroll Lock** key is used to toggle between **literal** and **command** mode. For example, choose ***Setup/Keys/Edit/ByName***. Select ***Place Pad***, which corresponds to the <F1> key. You should notice that the sequence of commands for this macro is: <ESC> <ESC> ***PP*** <ENTER> <ESC>.

The <ESC> and <ENTER> commands were entered by hitting <Scroll Lock> to put the program into literal mode and then hitting the <ESC> and <ENTER> buttons. This is the only way that these commands can be used in a macro. If you entered the characters "< E S C" etc. from the keyboard while in command mode and saved the result to the <F1> key the program will interpret it as follows:

a) The < character will be ignored.
b) The E character will be interpreted as the **Edit** command.
c) The S character will be interpreted as the **String** command.

and when <F1> is hit, you will be in **Edit String** mode. Not quite what you were expecting! The AUTOTRAX program is delivered with a selection of keyboard macros that approximate the keyboard commands of PCB version 3.

Users who have previously used the PCB program will find these macros very useful. New users have the option to change macros to suit particular needs. If you wish to change the default pad types or key macros, simply save your new pad or keyfile to TRAXEDIT.PAD

or TRAXEDIT.KEY. The next time you start the program, this file will be automatically loaded.

12. **Component Swapping**

The ability to swap a component to the other side of the PCB has been expanded to include Through-Hole Devices as well a Surface Mounted Devices. To swap component layers, use the *Edit/ Component/Layer* commands.

13. **Pads**

Pre-defined Pads have been reduced to 42. This allows the user to define up to 6 customs pads without having to make any deletions.

Edit Pad Global edit of Pads and Hole Size option *All Pads on PCB* is constrained toPads of the same shape, **and** with matching x and y sizes. Circular pads that have been edited from other shapes may carry hidden y dimensions that block Global edits.

14. **PCB3CON** (Conversion of PROTEL-PCB files to AUTOTRAX)

An option has been added to convert PROTEL-PCB v3.xx Dip Pads to Circular or Rounded Rectangle. Version 1.07 of PCB3CON required use of the *Edit/Pad/Global* command. Note that Dip Pads are converted to Circular 50 mils by PCB3CON. If you are using 12 mil tracks we recommend changing Pads to 62 mil Round.

Dip Pads are now converted to 50 x 120 instead of 40 x 120.

15. **Keep Out Layer & Board Layer differences**

The Keep Out Layer is used to define board outlines and no-go areas for the Auto Place and Autoroute. Lines on this layer are never crossed or touched by the router. Auto Place never places parts on these lines. Area Fills on this layer are left completely vacant by both Auto Place and Autoroute.

The Board Layer is for alignment targets, documentation, dimensions, text and any other items that the user doesn't want associated

with the artwork, but may want to plot with a given layer.

16. **Deleting all Free Primitives on a PCB**

In order to strip all free tracks and Vias, etc from a board without removing the components:

a) *Place* an Area *Fill* (top or bottom layer) which covers all the primitives you wish to delete.

b) Using *Highlight Net*, select the Fill (this may take some time on large board).

c) Select *Delete/Highlight*. This will delete all highlighted items except for components. Redraw screen **<END>**.

17. **Deleting Arcs**

In PCB Version 3, to delete an arc you had to find the center. In AUTOTRAX you need to place the cursor on the actual arc.

18. **Highlight/Connection**

To Highlight a connection, start on a track segment within that connection. Highlight/Connection works outwards until it reaches pads, which the Highlight will not pass. If you start from a pad, Highlight goes nowhere.

19. **The Bill Of Materials File.**

This file is generated by running the BOM.EXE file supplied on the PROGRAM DISK A. **Do not run BOM.EXE during an exit to DOS from either TRAXEDIT or TRAXPLOT.** A memory error is likely to occur if this is attempted.

20. **Component Library**

If you are working through the tutorial in Ch 3 "Laying out a PCB" you will be asked to place a component from the library by entering its name "DB25". Since the manual was written a number of DB25

family components have been added, ie: DB25/M, DB25/F, DB25RA/M. Enter the full name of one of these connectors and it will be placed successfully.

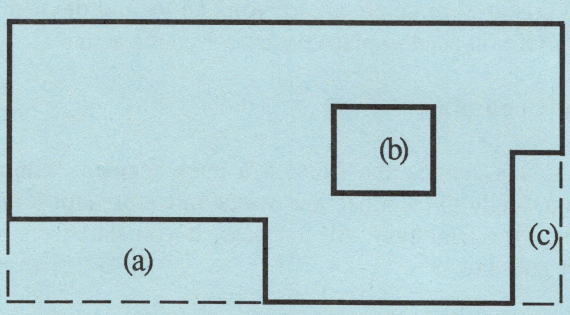
21. **New SMD components**

The components SOT23, SOT143 and SOT 89 have been added to the standard libraries.

AUTO PLACE Features

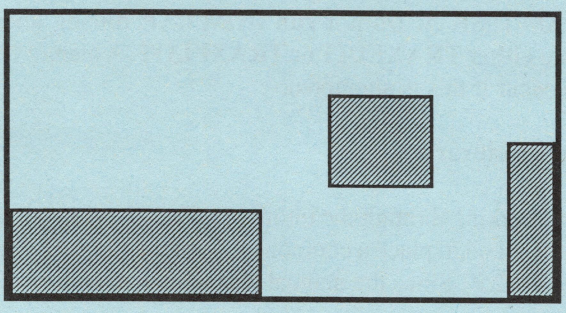
1. **Defining a non-rectangular board**

Before components can be placed, a board outline must be created in the Keep Out Layer. Using a simple line outline will not work when a non-rectangular board shape is required.



Keep Out
Tracks

In this diagram, Auto Place will put components in areas (a), (b) and (c). The correct way to define this board shape is:



Keep Out
Tracks

Keep Out
Area Fills

2. **Auto Place Setup**

An explanation of some of the features of *Auto Place/Setup* follows:

a) Large X & Y Placement Grid: Pin 1 (of components with 4 or more pins) will be placed on this grid.

b) Small X & Y Placement Grid: Pin 1 (of Bypass and Discrete components) will be placed on this grid.

c) Large X & Y Clearance: When 2 large components (of more than 3 pins) are placed, the distance between the outer edges of the 2 components including overlay layers) will be greater or equal to large clearances.

d) Small X & Y Clearance: Minimum distance between 2 small components or a small component and a large component.

3. **Bypass Capacitors, IC's and Power/ Ground Plane**

If you need to use Bypass Capacitors and Power or Ground Planes, some changes will be needed in order to successfully use Auto Place.

When Nets are loaded, the Power/Ground Nets are not stored, but converted to pad attributes. To use Auto Place, load Power and Ground Nets as normal Nets then run Auto Place (which will look after Bypass Capacitors). Clear Nets and reload them with the Power and Ground Planes assigned to net names.

4. **Auto Place Bypass Capacitor Rules**

To identify Bypass Capacitors, the program first finds all components with 4 or more pins. It then looks for 2 Nets that go to 70% of these components. The program then looks for all components with 2 pins, of which 1 connects to each of these 2 Nets. These are identified (by Auto Place) as Bypass Capacitors.

5. Auto Place Discrete Component Rules

Discrete components are those, other than Bypass Capacitors, that have less than 4 pins. They are assigned to large components by a simple rule. Ignoring the 2 identified power Nets, a Discrete component is attached to a large component if 50% or more of its pins are connected to the large component. So, a 3 pin Discrete requires 2 connections and a 2 pin "discrete" requires 1 connection.

These Discrete components are then grouped with the associated large component. The Bypass Capacitors are added and the whole unit is placed on the Large X & Y Grid. With Auto Place some trial & error is usually required to find a combination which is satisfactory. It is recommended that you save your layout before running Auto Place. That way, if you don't like the result you can reload the file and change the setup.

6. Multiple Libraries with Auto Place

The following procedure allows use of multiple component libraries with Auto Place.

a) Go to *Library/File* and select the first required library.

b) Run *Auto Place / Load Components*. Missing patterns will be reported at this stage.

c) Clear the Netlist (*Netlist/ Clear*).

d) Return to *Library/File* and specify new library.

e) Run *Auto Place / Load* again.

7. Re-running Auto Place

After components are placed by Auto Place they are "fixed in place". If no space can be found for a component then it is placed at the top of the board and set to "free to move". If you manually move a component then it becomes "fixed in place". The "free"/"fixed" status of a component can be changed using the *Edit/Component/*

Placement commands. When Auto Place is run, it treats "fixed" components as pre-placed and builds from them.

8. **Getting good results from Auto Place**

Auto Place locates components logically and conveniently to facilitate final placement.. Autoplace cannot locate edge connectors. Optimum layout of the board is left to the designer.

9. **Netlist Formats**

Netlists from schematic capture packages (other than PROTEL-SCHEMATIC) may conform generally to the PROTEL Netlist format. However, package names (DIP40, etc) and Pin identifiers may require editing to conform with AUTOTRAX standards.

OrCAD SDTIII® users can create compatible Netlists by using the Tango format option. **Tip:** the Key Field command ***KF,11 <Ret>,1 <Ret>*** presets Edit to receive AUTOTRAX library component descriptions after the reference designators and parts values are entered. Pin names have to be set to pin numbers. The OrCAD delimiter "," is accepted by AUTOTRAX INSTEAD OF "-". See the OrCAD Design Tools Addendum for further information.

10. **Transistors and other multi-pin discretes from Schematic Netlists**

A single Component pattern can have many different pin out designations. For example, a TO-92 can have various combinations of E, B and C. One solution is to leave the Schematic pin designations as E, B and C and then place components on the PCB. Use ***Edit/Pad*** to change the pad designator to the correct E, B or C. If you have a lot of pin outs in the same orientation, you may want to either make a special component in the library with the correct designations, or use ***Repeat Place*** after editing the first component.

11. **Move All Components onto Grid** has been added to the Auto Place Menu. This new feature allows you to set a **new grid size** and will shift pre-placed components onto the selected grid. We recommend using this feature before routing the board to insure that all components are on the same grid - a must for good router performance.

AUTOROUTER Features

1. **Single Layer Routing**

   Single Layer Routing has been added to the *Netlist/Route/Layer Setup* menu.

   a) Set the layer you wish to route to *Single Layer Routing.*
   b) Set all other layers to *Layer Not Used.*

2. **Smoothing Passes**

   Four "Smoothing" Passes have been added to the Route menu:

   a) Diagonal Remover: removes short diagonals which the Autorouter has duplicated over a single location.

   b) Loop Remover: removes half of an unintentional loop created by the Autorouter.

   c) Double Back Remover: attempts to replace needlessly indirect routes with straight track segments.

   c) Dual Stub Remover: attempts to reduce two short stubs connecting to a pad to a single stub.

3. **Two Via Remover** this routing pass attempts to remove one of two connected vias when they are within 100 mils of each other.

4. **Settings for SMD Routing**

Try:	Route Grid:	25 mil
	Route Tracks:	8 mil
	Route Via:	40 mil
	Track to Track:	8 mil clearance
	Pad to Pad:	8 mil clearance
	Pad to Track:	8 mil clearance

Check with your PCB manufacturer before designing to these specs. Set vertical and horizontal route layers depending on how the board

is organized. If you have an array of SMD memory chips arranged vertically on the top layer, then set the **Top Layer** to **Vertical.**

It is now possible to set a **5 mil Route Grid** for difficult SMD clearances. However, this option will slow the routing process considerably (each time you halve the grid pitch you double the number of potential paths for the router).

5. **Via Placement** can be controlled by placing Area Fills on an unused mid-layer wherever Vias are not wanted. (under ICs on double sided boards, etc). Because Vias pass through all layers of the board, they will not be placed in these filled areas.

6. **TRAXEDIT.LOG** (default file name) has been added which produces a text file report of Autorouter results, including a listing of any unrouted connections. The format for this report is:

```
PROTEL-AUTOTRAX STATISTICS LOG
LOG FILENAME  : TRAXEDIT.LOG
DATE          : 8-12-1988
TIME          : 13:21

INPUT  .PCB FILE : DEMOSMD.PCB
OUTPUT .PCB FILE : X BIAS.PCB

ROUTING PERFORMANCE
Pass                          Connections    Percentage    Time in
Title                           Routed       Completed       Pass
Process Pre-Routes     :                                   Not Used
Memory Routes          :          128          45.4        00:01:37
Horizontal             :            5           1.8        00:00:07
Vertical               :           14           5.0        00:00:19
L-Routes     (1 Via)   :           80          28.4        00:06:16
Z-Routes     (2 Via)   :           43          15.2        00:05:38
C-Routes     (2 Via)   :           10           3.5        00:04:07
Any Node     (2 Via)   :            1           0.4        00:01:33
Maze Routes (3 Via)    :                                   Not Used
Any Node     (3 Via)   :                                   Not Used
             Totals    :          281          99.6        00:19:37

SMOOTHING PERFORMANCE
Pass                          Tracks         Vias          Time in
Title                         Swapped       Removed          Pass
Via Minimization       :          63          53          00:04:01
Diagonal Remover       :           7           0          00:00:04
Loop Remover           :                                   Not Used
Double Back Remover    :                                   Not Used
Dual Stub Remover      :           1                       00:03:11
Two Via Remover        :           2           2           00:00:09

UNROUTED CONNECTIONS
Net Name: D7
          U5-15 to U1-19
```

7. **PROCESS PRE-ROUTES** pass in the router has been improved. You can now complete any part of any net and the router will reconnect around your work.

8. **DRC Clearances**

   The clearances in *Netlist/Route Separation* also apply to the Design Rule Check performed in *Netlist/DRC*.

9. **AUTOTRAX compared to PROTEL-ROUTE**

   Some former PROTEL-ROUTE users have wondered why AUTOTRAX seems slow by comparison. PROTEL-ROUTE is a simple channel router which has a fixed grid, fixed via size and fixed track size working in a batch mode. Hence the data structures are highly optimized. AUTOTRAX works with any grid size, track size, via size, user selected clearances, and unlike PROTEL-ROUTE, supports EMS and routes 6 layers.

   AUTOTRAX routes take longer to perform, but the final quality of the route is higher with less clean-up required. To improve Router performance:

   a) Make sure that you run *Netlist/Optimize* after using Auto Place, or after making significant changes to the layout of the placed board. This allows the program to adjust its routing strategy to the final position of components - resulting in a more direct path.

   b) Minimize the connection distances of discretes, such as resistors. The visible "ratsnest" that is present when you *Show/All Nets* is a good aid in repositioning components. Use the *Netlist/Length* command to track changes in the total connection length as you manually optimize the lay-out. A lower total connection distance will improve routing speed.

10. **NETCHECK program**

    This utility program, will be familiar to PROTEL-SCHEMATIC users. It generates a report comparing any two Netlists selected by the

user. From the DOS prompt, type **NETCHECK** to start the program. You will be asked for:

Schematic Net File Name> (include .NET extension)
PCB Net File Name> (include .NET extension)
Report File Name <NETCHECK.RPT> (default name)

The files must be present in the current directory. The text report generated will list all matching and non-matching Nets by name.

TRAXPLOT PROGRAM

1. Menu Duplications

a) From the Main Plot Menu select *Setup*. Notice the *Pens* selection. From the Setup menu select *Check Plot*. The Pens selection appears again. Press <ESC> or right mouse button. Select *Plotter* from the Setup menu and again notice that Pens is duplicated here. This has been done deliberately for your benefit, to make the program easier to use.

b) Similarly the *Options* menu is duplicated under *Setup/Plotter*, *Printer* and *Gerber*.

2. Reference Manual Changes.

a) On pg. 165 of the Reference Manual, the *Pad Hole Size* selection in the *Options* menu is now called *Pad Hole Guide Size*. The size entered here is used only as a guide for drilling. The actual size of the hole to be drilled is stored in Pad Hole Size in the TRAXEDIT program.

b) The *Setup* Menu on pg. 166 now contains the *Pens* selection between NC Drill and Menu colors.

c) On pg. 169, in the Gerber Setup menu, the *Plot Tolerance* option has been removed from the program. Two new additions are *Arc Quality* and *Options*.

3. **The Plotter Setup Menu** (pg. 167)

   Some plotters have in-built arc drawing algorithms. Arcs drawn in this way are know as **Hardware Arcs.** Alternatively, if your plotter does not have this capability, the computer must calculate points on the arc and drive the plotter. This is a **Software Arc.** The Plotter Setup Menu gives you the option of using either method with the **Software Arc** selection. The *Arc Quality* selection is used if Software Arc is set to ON. It allows the user to set the length of the chord (in mils) which will be used to draw arcs. This option is particularly useful for arcs on the Overlay layer. The default settings are OFF and 5 respectively. If the plotter does not have Hardware Arc capability and Software Arc is set to OFF, the program will recognise this and use the Software Arc option.

4. **NC Drill Tool files** Name your .TOL file "Drill.Tol" then it will be loaded at Startup. In version 1.07 you weren't able to load it manually. This restriction has been removed.

5. **Tracks on Pad Centers** Tracks are automatically backed away from the Pad centers in order to plot pad holes (if selected in Plot/Setup). For a track to be backed away from the center of a pad, the track must end within 10 mil (in X and Y) of the pad center. In PCB 3, the track had to be exactly on the pad center. In Autotrax the pad centers are no longer important.

6. The **Gerber Setup** Menu (pg. 169) contains the Arc Quality selection as AUTOTRAX does not support the Gerber format Hardware Arc.

7. The **Setup Pens** Menu contains two selections:

   a) **Check Plot Pens**
   b) **Plotter Pens**

   Each contains menus for **Pen Allocation**, **Pen Speed** and **Pen Sizes.**.

   Choose *Setup/Pens/Plotter Pens*. Toggle the *Main Pen* selection through its possible values. These are 1, 2, 3, 4, 5, 6, 7, 8, AUTO. Each of the numeric settings mean that the program will select that number pen. On AUTO, the program, will select a pen for each

primitive on the board (track, pad, etc), based on the user defined **Pen Sizes.** If a Pen Size is set to 0 the program assumes that the pen is not being used. When the Main Pen is assigned a value between 1 and 8, TRAXPLOT automatically sets the appropriate Pen Size to 30 if it was previously 0. This guards against a 0 pen size setting for a selected pen.

Hit <ESC>/Right Mouse Button until you are back to the Setup Pens Menu. Choose *Check Plot Pens*. Choose ***Pen Allocation***. When you select a layer the pen value will change. If a layer is assigned a particular pen then the following checks are done. First, a check is made to see if the layer is ON or OFF. If the layer is ON then the Pen Size, for the particular pen, is set to the default value of 30. If the layer is OFF the Pen Size remains 0.

IMPORTANT

There is quite a difference between PROTEL-PCB version 3 PLOT and TRAXPLOT with regard to pen sizes for 1:1 and 2:1 plots. In the PCB PLOT program it was necessary to use a .6 pen for 2:1 and .3 for 1:1. With TRAXPLOT it is not necessary to change the pen size for different SCALES. While it is recommended that a .6 pen is used for 2:1 and .3 for 1:1 plots, the program will calculate the correct number of passes for each pen size and scale. If gaps occur between pen passes it is likely that the pen size has been specified larger than it physically is.

It should be noted that all pen sizes are measured in MILS. (1 mil = 0.001 inch). Thus a standard .3 millimeter size pen would be entered as a 12 mil size pen. (0.3 mm = 0.012 inch).

The quality of final plots is determined by three factors, namely the pen, ink and film types. In particular, different combinations of ink and film can result in different line thicknesses being drawn with the same pen. As a result there may be either, small gaps between pen passes caused by thinner than expected lines, or slightly thicker than expected lines causing pad hole fill-ins. In the first case it would be necessary to experiment with pen size values slightly larger than the actual physical size. In the second, smaller than actual size values would be needed.

8. **The NC Drill Menu** selection is found at the Main Menu level. It operates in a very similar way to the Gerber Plot menu. Tool numbers and drill sizes stored in the Tool Table are matched to pad hole sizes, within a specified tolerance level. Again these parameters are defined in the *Setup/NC Drill* menu.

   Upon completion of matching, two files are written to disk, namely .DRL and .TXT files. The .DRL file is in Excellon format. The .TXT is its ASCII equivalent.

9. **Calcomp Plotter Driver** will work for both non-intelligent and intelligent Calcomp plotters using the 960 format. At present, software arcs are used but the next version will make use of the plotters' inbuilt arc facility which will increase the speed of the plot.

   The communications menu of the plotter must be setup as follows:

0	Parity & Bits/Char	Parity = 0 (none)	Bits/Char = 8
1	Stop Bits	Stop Bits = 1	
2	Interface	Interface = 0 (Serial)	
3	Host Baud & Mode	Host Baud = 2400	Mode = 1 (960)
4	Muting & Checksum	Term Muting = No Checksum enable = No	
5	Isochronous & EOM	Isochronous = No	EOM = 03
6	Direct Ctrl & XON/XOFF	Direct Ctrl = Yes XON/XOFF = No	

10. **Confusion over pen sizes in TraxPlot**

    In TraxPlot the pen size specifies the width of the line that the pen draws. This measurement is in mils (.0001 inch). If using the pens from PCB 3 the 0.3mm pen should be set at 13 mil and the 0.6mm pen set at 26 mil. Don't use 30 and 60 mil for these pen sizes.

Note also that the Pad Hole Guide Size, Solder Mask and Paste Mask enlargements are also in mils, not metric.

Software Arc Chord Length is also measured in mils. At least 6 segments are now used in a circle, no matter how large the chord is set.

11. **New option in TraxPlot** The user can now turn off Vias in any Plot or Print from the *Option* menu, under the main Plot menu.

12. **The Gerber Plot Menu**

AUTOTRAX allows the creation of Gerber format photoplot files. This requires that the operator have some knowledge of the target photoplotter. Specifically, a list of apertures defining the size shape and usage of each must be provided. From DOS, enter the command **TYPE PRECIS1.APT <RET>** for an example. This file includes an explanation of the format. If you intend to use a photoplotting bureau, it is necessary to obtain aperture file format information directly from them.

From the TRAXPLOT Main Menu select *Setup/Gerber*. After providing the relevant pcb filename and selecting appropriate X and Y offsets (often 0.000) the appropriate Aperture Table must be selected. It is important to realize that the apertures available may not exactly match the pads described by the PCB file.

TRAXPLOT will automatically assign apertures to match pads within the **Match Tolerance**. In general the match tolerance should not be too liberal as clearance errors may result. If unsure then use a Match Tolerance of 5% or less.

G54 should normally be **OFF**. This is required by some of the older plotters to precede aperture change commands.

Circles and arcs are drawn as a series of chords. **Arc Quality** determines the chord length, in mils. The smaller the number (eg 5) the better the circle approximation, and the longer the process will take.

From the *Options* menu set the type of plot (eg Top Layer) turn off Border, Title Block, Vias in Solder Mask (normally) and set Pad Hole Guide Size (not applicable to photoplotting), Solder and Pad Mask Enlargements to 0. The Solder and Pad Mask enlargements are only applicable to mask type plots. The enlargement percentages can be recommended by your PCB manufacturer.

Return to Main Plot Menu and select *File*, then *Load*. Supply your PCB filename. The file will be checked then loaded. Progress information is displayed.

Return again to the Main Plot Menu and select *Gerber Plot*. At this point TRAXPLOT will assign draft codes from the aperture table to objects from the PCB file. The **Match Tolerance** selection allows the user to set the level of matching. This setting instructs the program to match draft codes with pad sizes which are approximately of the same area, within the percentage error set. If no aperture can be found within the Match Tolerance specified, the operator will be asked for assistance. TRAXPLOT will provide Flash, Stroke and Fill options, or you can force some other solution by selecting manual assignment. This allows the choice of any draft code and any method. A summary of matches made is provided in a text file the name of which is presented to the operator (.MAT).

The layer file can now be generated. Progress information is displayed during this process. To create another layer file one simply changes the type of plot in *Setup Options*, then selects *Gerber Plot* from the Main Plot Menu. There is no need to Reassign Draft Codes although the choice is there.

In general if one knows that a particular photoplotter is to be used it is a good idea to create a TRAXEDIT component library corresponding to the apertures available on that plotter. In this way 100% match is always assured thus avoiding any possible tolerance errors and speeding up the plot by obviating the necessity for time expensive fills.

© Copyright 1989 PROTEL TECHNOLGY PTY LTD
Rev 16.1.89